THEORETICAL & APPLIED MECHANICS LETTERS **3**, 062001 (2013)

Current studies on air distributions in commercial airliner cabins

Wei Liu[1,2] and Qingyan Chen[2,1,a)]
[1)] *School of Mechanical Engineering, Purdue University, West Lafayette 47907, USA*
[2)] *School of Environmental Science and Technology, Tianjin University, Tianjin 300072, China*

(Received 29 October 2013; accepted 6 November 2013; published online 10 November 2013)

Abstract Air distribution in commercial airliner cabins is very important for the comfort and health of passengers and crew. Experimental measurements, computational fluid dynamics (CFD) simulations, and inverse modeling are state-of-the-art methods available for studying the air distribution. This paper gave an overview of the different experimental models, such as scale models, simplified models, full-scale mockups, and actual air cabins. Although experimental measurements were expensive and time consuming, the data were essential for validating CFD simulations. Different modeling strategies for CFD simulations were also discussed in this paper, including large eddy simulations and Reynolds averaged Navier–Stokes equation modeling. CFD simulations were main stream approaches for studying the air distribution but they could not easily lead to optimal design. Inverse modeling of air distribution has recently emerged into a very powerful and attractive tool for designing the air distribution in airliner cabins, although most of the studies were preliminary.
 [doi:10.1063/2.1306201]

Keywords aircraft cabin, airflow, experiment, CFD, inverse modeling

I. INTRODUCTION

Air distribution in commercial airliner cabins mainly has two functions: regulating air temperature and air velocity to create a thermally comfortable environment and providing adequate ventilation for reducing gaseous and particulate contaminant concentrations for maintaining a safe and healthy environment.[1] Although the aerospace industry has improved the environmental control systems to ensure the comfort levels and hygiene in aircraft cabins in the past decades,[2] there are still reports of discomfort and health problems.[3] Therefore, it is necessary to further study air distribution in the airliner cabins in order to design better environmental control systems.

There are mainly two methods available in studying the air distribution in an aircraft cabin: experimental measurements and numerical simulations by computational fluid dynamics (CFD). The experimental studies are usually thought to be more reliable but they are often expensive and time consuming. Today very few experimental measurements were used to study the air distribution. Instead, most of the measurements were to obtain high quality data for validating numerical simulation results.

The numerical simulations by CFD are cheaper and more informative compared with the experimental measurements. However, the numerical models used approximations and simplifications in modeling the flow so the simulation results were of uncertainties. Thus, the best approach is to use the experimental data to validate the CFD tool and the validated CFD tool is used to study air distribution so the simulated results can be trusted.[4,5]

Although CFD can be used to design air distribution, the design may not be optimal. Recently, the research community has started to look for CFD based optimal design by using inverse modeling. The inverse simulations, although the results are preliminary, have shown great potential for optimal design of air distribution for airliner cabins.

This paper is to provide a brief overview of the experimental measurements, numerical simulations by CFD, and inverse modeling of air distributions for airliner cabins.

II. EXPERIMENTAL MEASUREMENTS OF CABIN AIR DISTRIBUTIONS

Experimental measurements of air distribution in airliner cabins can be conducted in a cabin mockup or in an actual airplane. Cabin mockup includes scaled model, simplified model and full scale model.

For example, Poussou et al.[6] conducted experimental measurements on a one-tenth scale, water-based empty cabin model for investigating the effects of a moving human body on flow and contaminant transport inside an aircraft cabin. The scale model was inexpensive and can obtain high-quality data. When conducting experimental measurements of air distribution in airliner cabins, it is essential to ensure flow similarities. The air distribution is driven by the inertial force from the diffusers and buoyancy forces from the thermal plumes generated by the passengers. Thus, the Reynolds number (Re) and Grashof number (Gr) must be the same as those in actual air cabins. Even with a different fluid, it is hard to achieve the same Re and Gr. Therefore, the flow features in a scale model would differ from that in a real aircraft cabin.[7] Therefore, scaled model was not popular at present.

[a)] Corresponding author. Email: yanchen@purdue.edu.

A simplified model has the same character length as a real aircraft cabin, such as cabin height. But, the geometry is greatly simplified. For instance, Wang and Chen[8] measured the airflow and temperature fields in a cubic box with 2.44 m in each dimension as shown in Fig. 1. The cubic box could represent a half section of a twin-aisle airline cabin of Boeing 767 with three rows of seats and 10.5 passengers inside (21 passengers for the whole cabin). The passengers were simulated by a heated box as shown inside the box. A linear diffuser was located on the left wall near the ceiling and an exhaust slot was located near the floor on the right wall that was similar to Boeing 767. Due to the simplifications, the experimental measurements can provide high quality flow and temperature data. However, due to the absence of geometric similarity, it is hard to convince that the flow in the simplified cabin can actually represent that in an actual cabin.

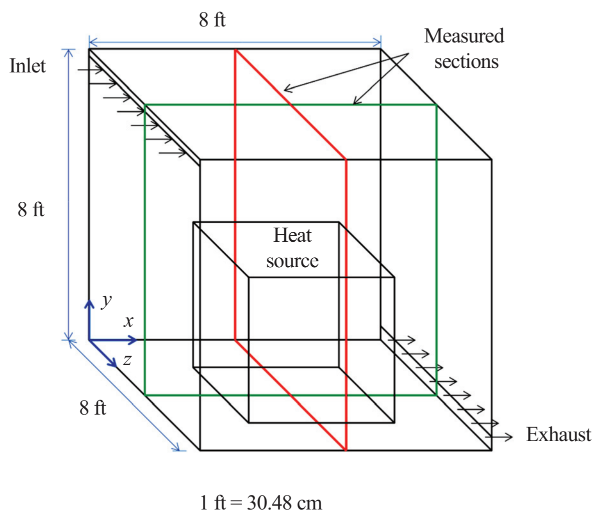

Fig. 1. A simplified air cabin model used by Wang and Chen.[8]

The main stream research in cabin experimental measurements used a section of a full-scale model that has the same shape and size with real aircraft cabins. In addition, the thermo-fluid boundary conditions were also similar to those in real aircraft cabins. Figure 2 shows a full-scale Boeing 767 aircraft cabin mockup built by Zhang et al.[9] The cabin mockup had four rows with 28 passenger seats and 14 of them were with heated boxes. Due to the difference of the duct system between the mockup cabin and actual Boeing 767 airplane, the corresponding flow features may still be different. Nevertheless, it is easy to control the thermo-fluid boundary conditions and to change the cabin interior setting, such full-scale cabin mockup models are the most popular ones for studying air distributions in airliner cabins.

Very few experimental studies of air distribution in airliner cabins were conducted in actual airplanes. Tianjin University[10] has recently used a functional MD-82 plane for such study. Figure 3 shows that they have used the three-row, first-class cabin with heated manikins for studying the air distribution. The thermal manikins built by wrapping them with nickel-chromium wires were used to model the seated passengers. Although the airplane was functional, the thermo-fluid boundary conditions were difficult to control. For example, we found that the airflow rate and velocity from the diffusers along the longitudinal direction were not uniform. The airplane was designed to have minimal longitudinal flow but the experimental measurements showed very significant longitudinal flow. It is very difficult to measure accurately the air distribution. The experimental data quality suffered a lot, when Liu et al.[11] used the data to compare the performance of the *Re*-normalization group (RNG) k–ε model[12] and large eddy simulation (LES).[7] The errors in the experimental data were comparable to those between the two CFD models. Thus, such data may not be suitable for validating a CFD tool.

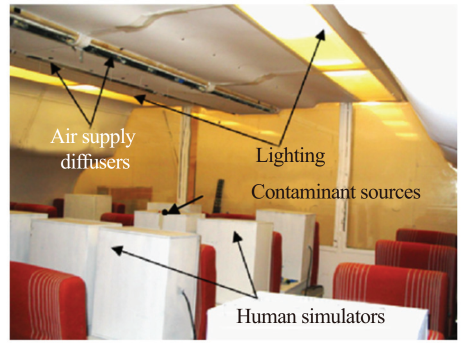

Fig. 2. A section of a full-scale aircraft cabin mockup built by Zhang et al.[9]

Fig. 3. The first-class cabin with thermal manikins in a functional airplane at Tianjin University.[10]

III. CFD SIMULATIONS OF CABIN AIR DISTRIBUTIONS

Wang and Chen[8] had evaluated eight CFD models for their ability to simulate air distribution in airliner cabins. The CFD models used were the indoor zero-equation model (0-eq),[13] Launder–Sharma model (LRN),[14] RNG k–ε,[12] shear stress transport (SST) k–ω model,[15] v2f model (v2f),[16] Reynolds stress model (RSM),[17] LES with dynamic Smagorinsky subgrid-scale model (LES-DSL),[7,18] and detached eddy simulation (DES)[19] that was the combination of LES and spalat-allmaras model[20] (DES-SA). The experimental data in the simplified model was used for the evaluation of the CFD models.

Figure 4 shows the performance of the eight models in predicting the air velocity in position 6 in the cubic box as shown in Fig. 1. Position 6 is typical as the agreement of numerical results and experimental data at this position is average. Their study also compared the turbulence kinetic energy and air temperature profiles at position 6. The results show that the LES-DSL was the best among all the models. Besides, the v2f model performed the best among the Reynolds averaged Navier–Stokes equation (RANS) models. The RNG and RSM models showed similar and acceptable accuracy in predicting the velocity, turbulence kinetic energy, and air temperature. These four models gave acceptable accuracy in predicting air distribution in aircraft cabins.

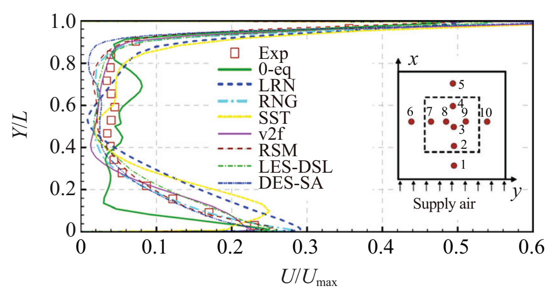

Fig. 4. Air velocity profiles predicted by the eight turbulence models at position 6 in the cubic box.[8]

Besides the accuracy of the CFD models, the efficiency of these models is also very critical in designing air distribution. The computing time needed for these four CFD models could be expressed as

$$t_{\mathrm{LES}} \gg t_{\mathrm{RSM}} > t_{\mathrm{v2f}} \approx t_{\mathrm{RNG}}.$$

The LES required a computing time at least two orders of magnitude longer than the RANS models. This is because one has to solve the transient flow even if the flow is steady and the LES requires a grid of higher resolution. The RSM explicitly solves the transport equations of Reynolds stresses and fluxes instead of calculating turbulence eddy viscosity, so it requires more computing time than the v2f and RNG, which assume isotropic turbulence structures. Theoretically, the v2f requires more computing time than RNG because it calculated additional $\overline{v'^2}$ transport equation, the fluctuation of normal velocity. In fact, the difference in computing time of these two models was minimal. For design and study of air distributions in airliner cabins, the mean air parameters are more useful than turbulent parameters.[21] So the RANS modeling is preferred and the v2f is highly recommended.

IV. INVERSE MODELING OF CABIN AIR DISTRIBUTIONS

Validated CFD tools are now widely used for studying air distribution in airliner cabins for better thermal comfort and air quality. The CFD tools would calculate the distributions of air velocity, air temperature, and contaminant concentrations in the cabins with thermo-fluid boundary conditions. If the study is for design, it would take tens or even hundreds calculations to reach a good design. Even through the design may not be optimal. Therefore, recent trend on studying air distribution is to use CFD based inversing modeling. The inverse modeling uses desirable thermal comfort and indoor air quality or the corresponding air velocity, air temperature, and contaminant concentration distributions in cabins as the design objective. The modeling strategy is to identify the thermo-fluid boundary conditions for achieving the design objective.

Early work conducted by Zhang and Chen[22] was to combine CFD with a quasi-reversibility equation to inversely identify the contaminant source in a Boeing 767 aircraft cabin mockup. The quasi-reversibility approach reversed the time marching direction of the governing transport equation for the contaminant transport. With the contaminant distribution at $t = 16$ s in a section of the cabin as shown in Fig. 5(a), the quasi-reversibility approach can calculate the contaminant source location at $t = 0$ s as shown in Fig. 5(b). The actual source was a point one but the calculated source was dispersive. Nevertheless, the approach can approximately identify the contaminant source location.

Xue et al.[23] integrated the CFD and genetic algorithms in finding the optimal thermo-fluid boundary conditions in achieving desirable thermal comfort in the first-class cabin of an MD-82 aircraft as shown in Fig. 6. They evaluated the thermal comfort by predicted mean vote (PMV) and set the desired PMV around the passengers between −0.1 to 0.1. They found that multiple solutions existed for satisfying the PMV. Figure 7 shows the inlet air velocity and temperature relationship that can produce the required PMV in the aircraft cabin.

Recently we have used adjoint method to achieve optimization in design. The approach begins with an initially guessed inlet thermo-fluid boundary conditions. A set of state equations (Navier–Stokes equations) were solved with this initial inlet boundary conditions to see if the objective function (optimal design criteria) could be satisfied. If not, the approach computes the sensitivity of the objective function with respect to the variation of the inlet boundary conditions. Then a descent method can be used to determine the change of design variables that can decrease the objective function. Finally a set of adjoint equations are solved to obtain the inlet boundary conditions. Figure 8 shows the optimization process for a half of the first-class airliner cabin as shown in Fig. 3. A cycle is an iteration of the above-mentioned method that is automatically excuted during the computer simulation. Cycle 0 was PMV distribution with the initial boundary conditions. The PMV

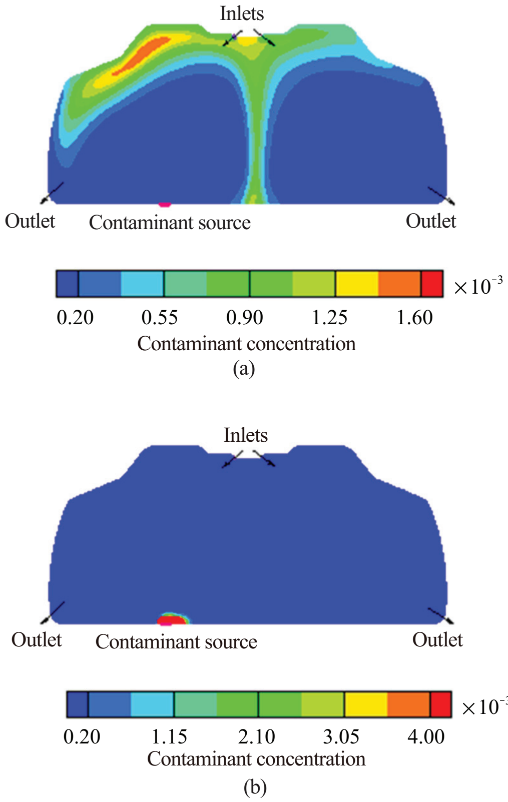

Fig. 5. (a) The initial contaminant concentration distribution at $t = 16$ s in a section of an airliner cabin and (b) the contaminant source location identified by the quasi-reversibility approach at $t = 0$ s.

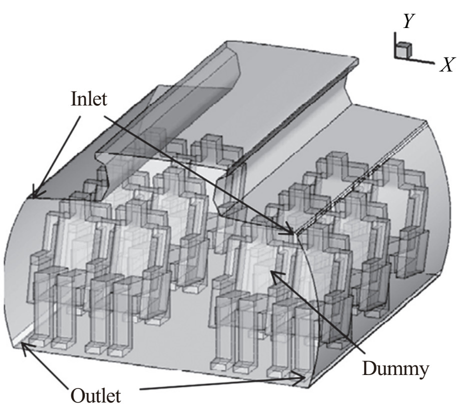

Fig. 6. The first-class aircraft cabin.

was too high from thermal comfort point of view. After 42 cycles, the PMV falled between −0.2 and 0.2 that was within the design objective. The adjoint method can achieve the design objective through one calculation, not tens or hundreds calculations as CFD does. Thus, the inverse design is powerful and can reduce significantly human effort.

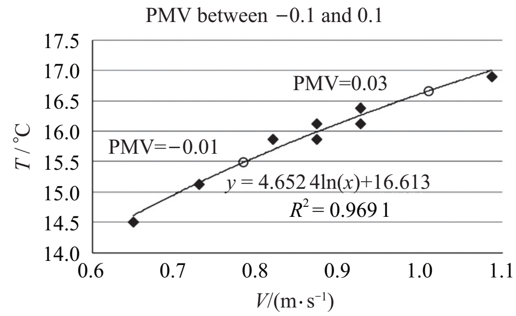

Fig. 7. The inlet air velocity and temperature relationship for satisfying the PMV in the cabin.

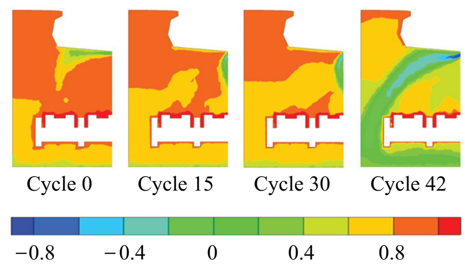

Fig. 8. The PMV distribution in a section of the first-class cabin of an airliner cabin through the optimal design process by the adjoint method.

V. CONCLUDING REMARKS

This paper discussed the recent status in studying air distribution in commercial airliner cabins. Experimental measurements are still regarded as the most reliable method. Among the scale-models, simplified models, full-scale models, and actual airplane cabins, full-scale models are most widely used due to its control on thermo-boundary conditions. The scale models may not show the same flow features as those in actual cabins, while the flows in actual cabins are too complex. The simplified models can produce high quality data, if one believes the flow is similar to that in an actual cabin.

CFD tools with LES and RANS models are widely used for studying the air distributions. Among many popular turbulence modeling approaches, the LES-DSL was the most accurate model but the computing time was very long. The v2f model performed the best among the RANS models, while the RNG k–ε model was sufficiently good. RANS models need much less computing time so they are more acceptable in engineering applications.

Recent trend was to use inverse modeling strategies. Preliminary results have shown that different inverse models could be used to identify contaminant source location, to design cabin environment with good thermal comfort, etc. The beauty of inverse modeling is that it can achieve optimal design through one single calculation, although multiple solutions could exist. In addition, air distribution is not the only subject for cabin environment design. The studies on infectious disease transmissions, multiple-factors on cabin environment, uncertainties and human factors, etc., are receiving more attentions today than ever.

This work was supported by the National Basic Research Program of China (973 Program) (2012CB720100).

1. Committee on Air Quality in Passenger Cabins of Commercial Aircraft, Board on Environmental Studies and Toxicology, Division on Earth and Life Sciences, National Research Council, *Airliner Cabin Environment and the Health of Passengers and Crew* (National Academy Press, Washington DC, 2002).
2. R. P. Garner, K. L. Wong, S. C. Ericson, et al., *CFD validation for contaminant transport in aircraft cabin ventilation flow fields*. Proceedings of annual SAFE Symposium on Survival and Flight Equipment Association (2003).
3. K. Leder and D. Newman, Internal Medicine Journal **35**, 50 (2005).
4. H. Mo, M. Hosni, and B. Jones, ASHRAE Transactions **109**, 101 (2003).
5. T. Mizuno and M. J. Warfield, ASHRAE Transactions **98**, 329 (1992).
6. S. B. Poussou, S. Mazumda, M. W. Plesniak, et al., Atmospheric Environment **44**, 2830 (2010).
7. M. Germano, U. Piomelli, P. Moin, et al., *Dynamic subgrid-scale eddy viscosity model*. U.S. Department of Transportation, DOT/FAA/AM-04/7 (1996).
8. M. Wang and Q. Chen, HVAC & R Research **15**, 1099 (2009).
9. Z. Zhang, X. Chen, S. Mazumdar, et al., Building and Environment **44**, 85 (2009)
10. W. Liu, J. Wen, J. Chao, et al., Atmospheric Environment **56**, 33 (2012).
11. W. Liu, J. Wen, C. H. Lin, et al., Building and Environment **65**, 118 (2013).
12. V. Yakhot and S. A. Orszag, Journal of Scientific Computing **1**, 3 (1986).
13. Q. Chen and W. Xu, Energy and Buildings **28**, 137 (1998).
14. B. E. Launder and B. I. Sharma, Letters in Heat Mass Transfer **1**, 131 (1974).
15. F. R. Menter, AIAA Journal **32**, 1598 (1994).
16. L. Davidson, P. V. Nielsen, and A. Sveningsson, Turbulence, Heat and Mass Transfer **4**, 577 (2003).
17. M. M. Gibson and B. E. Launder, Journal of Fluid Mechanics **86**, 491 (1978).
18. D. K. Lilly, Physics of Fluids **4**, 633 (1992).
19. M. Shur, P. R. Spalart, M. Strelets, et al., *Detached-eddy simulation of an airfoil at high angle of attack*. Proc. of 4th Int. Symposium on Eng. Turb. Modeling and Experiments, Corsica, France (1999).
20. P. Spalart and S. Allmaras, *A one-equation turbulence model for aerodynamic flows*. Technical Report AIAA-92-0439 (1992).
21. Z. Zhai, Z. Zhang, W. Zhang, et al., HVAC & R Research **13**, 853 (2007).
22. T. Zhang and Q. Chen, Indoor Air **17**, 439 (2007).
23. Y. Xue, Z. Zhai, and Q. Chen, Building and Environment **64**, 77 (2013).